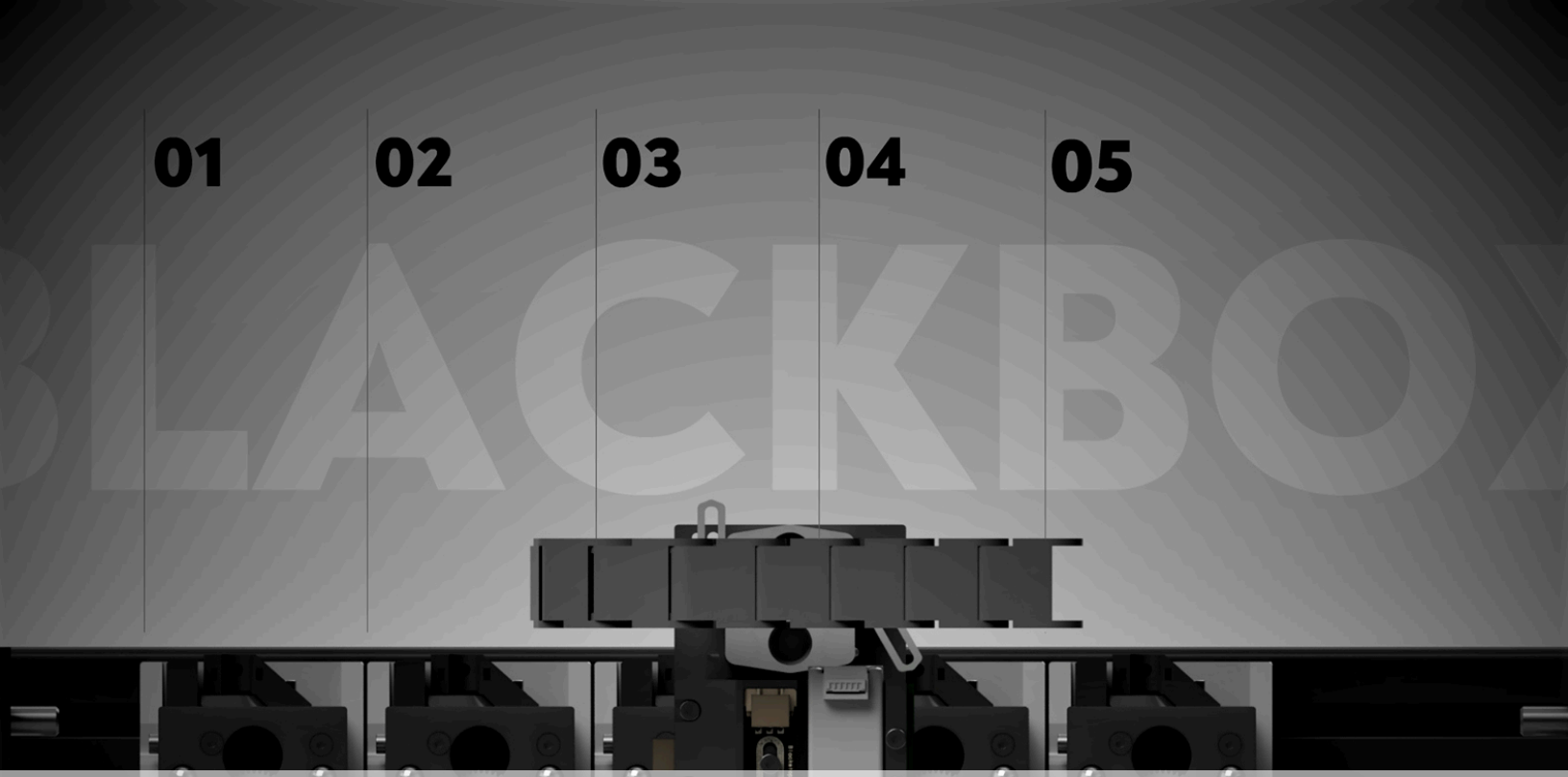


BLACKBOX CE

Printed Parts Guide + Supplements

DOCUMENTATION VERSION 1

Blackbox CE Getting Started Guide

Change Log

Version	Notes
1	Initial Release

BEFORE YOU BEGIN

Nozzle:

A 0.4mm nozzle is recommended for most parts, and some smaller parts were designed with this extrusion width in mind . If using a larger nozzle, ensure the resulting parts are strong and clean and accurate. Support settings and XY size compensation (AKA Horizontal expansion) should be tuned for your material prior to part printing.

Material:

Considerations include rigidity, heat deformation resistance, and stress creep resistance. Most materials are viable (aside from PLA) Best results will come from a glass or carbon reinforced polymer. If a non-filled material is used, stay away from the more elastic plastics like PETG.

Print Settings:

For best strength, print on the hotter end of your material temperature range. Use a low (or zero) part cooling fan speed where possible. Consider using the part cooling fan only on overhangs, support interfaces, and bridges.

General Guidelines:

Blackbox parts are designed with printability in mind. Most parts will have an obvious orientation preference, and most are smaller in size. Still, printing in an enclosed build volume can help mitigate any warping with some materials.

Spend some time perfecting the profile used for your chosen filament. Pay close attention to any "Elephants Foot" during test prints. Also ensure that walls are flat with no "pimples" from over extrusion. Consider enabling and tuning linear advance to mitigate bulging sharp corners.

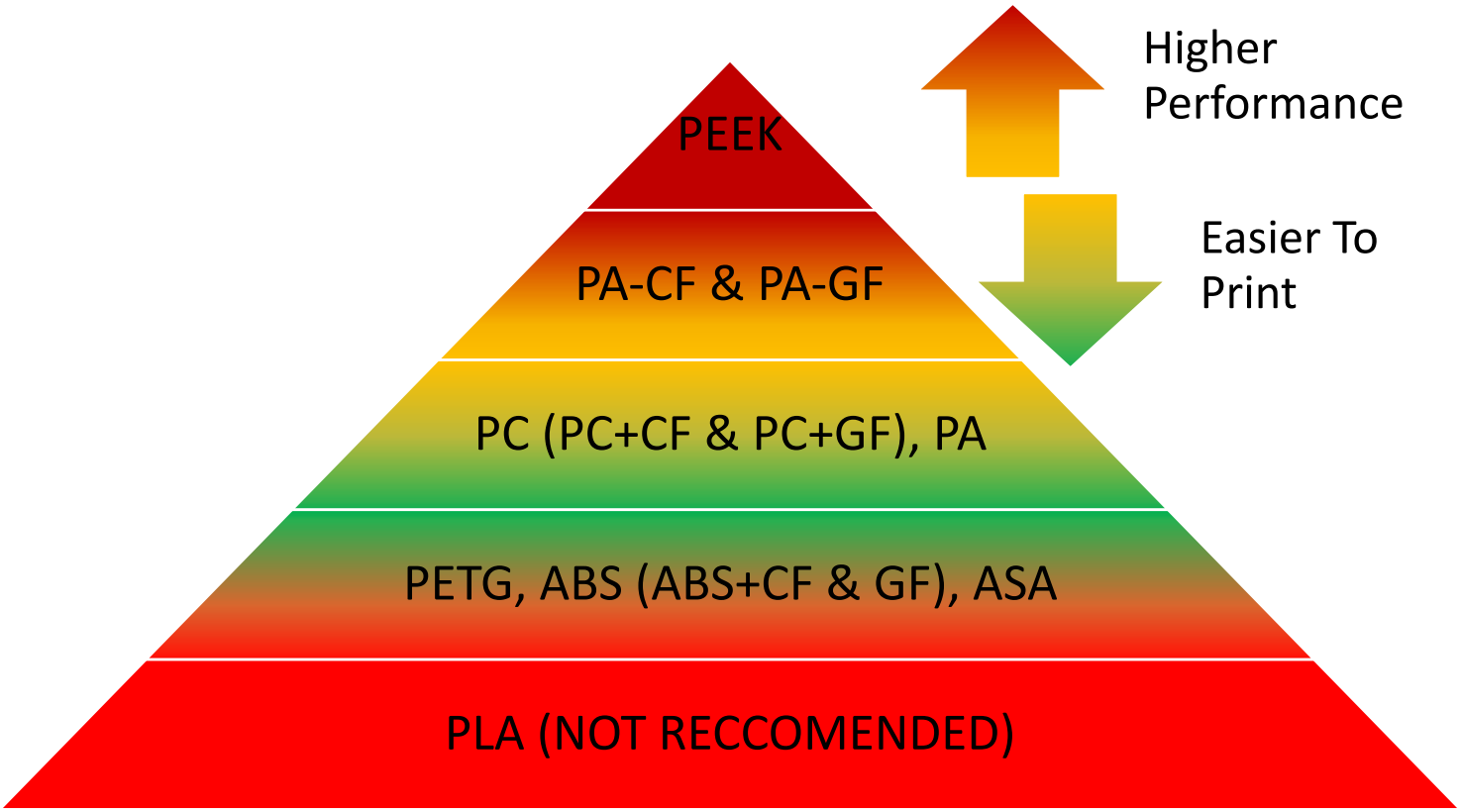
Blackbox has many printed parts and will require many days of printing along with 1.5-2.5 KG of material. The actual time and material amount will depend on your specific build.

Materials & Settings

Below are the recommended specifications for FDM printing of Blackbox parts. All slicing programs will have these settings with slight variations in naming.

- **LAYER HEIGHT: 0.2mm**
- **LINE WIDTH: 0.4mm**
- **INFILL: 40-50% (Grid, Gyroid, Honeycomb, Triangle or Cubic)**
- **PERIMETERS: 5-6**
- **TOP & BOTTOM SOLID LAYERS: 5-8**

Material Options



What we recommend:

Carbon Fiber Filled PolyCarbonate (ABS Blend OK)

Why?

PC is a capable engineering material that can stand the test of time under the stresses of tightened fasteners and belts. The Carbon fiber adds rigidity to what would otherwise be a material prone to flex. The fiber fill also helps to avoid warping during/after printing and allows for cleaner bridging and overhangs.

Tuning Your Filament Profile

As stated above, you should be sure to spend the time getting to know your filament of choice and how it interacts with your printer. Due to the diversity of filaments and machines, there is no one-size-fits-all standard that we can supply you with. Double check the performance of the following settings to have the best experience with the least amount of post- processing.

1. **Temperature**
2. **Retraction**
3. **Print speeds and accelerations/jerk**
4. **Horizontal Expansion (XY Size Compensation)**
5. **Extrusion E-steps / Flow Rate**
6. **Support Z distance**
7. **Support density and interface**
8. **Bridging speed and flow rate**
9. **Linear Advance**

Check out this lovely and free guide from TeachingTech for some helpful tips/tools on many of these topics.

Blackbox Printed Parts Readiness Test

Use this guide to assist in calibrating your printer and material in preparation for printing your Blackbox Toolchanger parts.

Download the STL here!

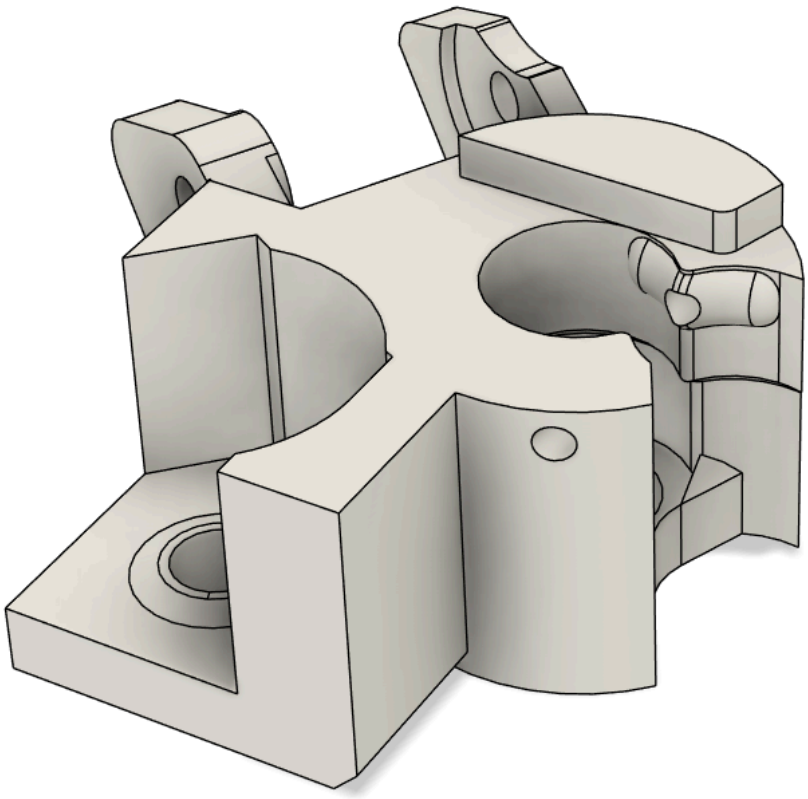
To confidently complete this guide, you will need a pair of measuring calipers (digital is fine) with a displayable resolution of at least 0.1mm

Introduction:

This guide will focus on the real-world results of your printer, slicer settings, and material. This test should be performed after most of your settings are well tuned. Before getting started here, print some calibration cubes, heat towers, retraction test pillars, ETC.

Once you are happy with the overall performance and quality of your printer and material combination, follow this guide to print your Blackbox Readiness Test Part (or BRTP for acronym enthusiasts)

A look at what we are printing:



Overview:

This test print has been purpose designed to include the most challenging and critical features you will encounter while printing your parts. It includes the following features/challenges that you will find throughout your printing process:

1. **Sharpest non-supported overhang present on Blackbox**
2. **Various vertical hole positions**
3. **Shaft/Bore tolerance locations**
4. **Supported overhangs**
5. **Critical X, Y, and Z dimensions**
6. **High retraction counts**
7. **Flat and curved surfaces**

General Slicing Settings:

Wall/Perimeter Count: 4

Number of Bottom Layers: 3

Number of Top layers: 3

Infill: 20%

Extrusion width: 0.4mm (Use only a 0.4mm nozzle for this print!)

Supports: None

Remember:

This guide will focus on confirming the results of your printer and profile tuning. It will offer some specific tips along the way but is not meant to be a comprehensive guide for tuning.

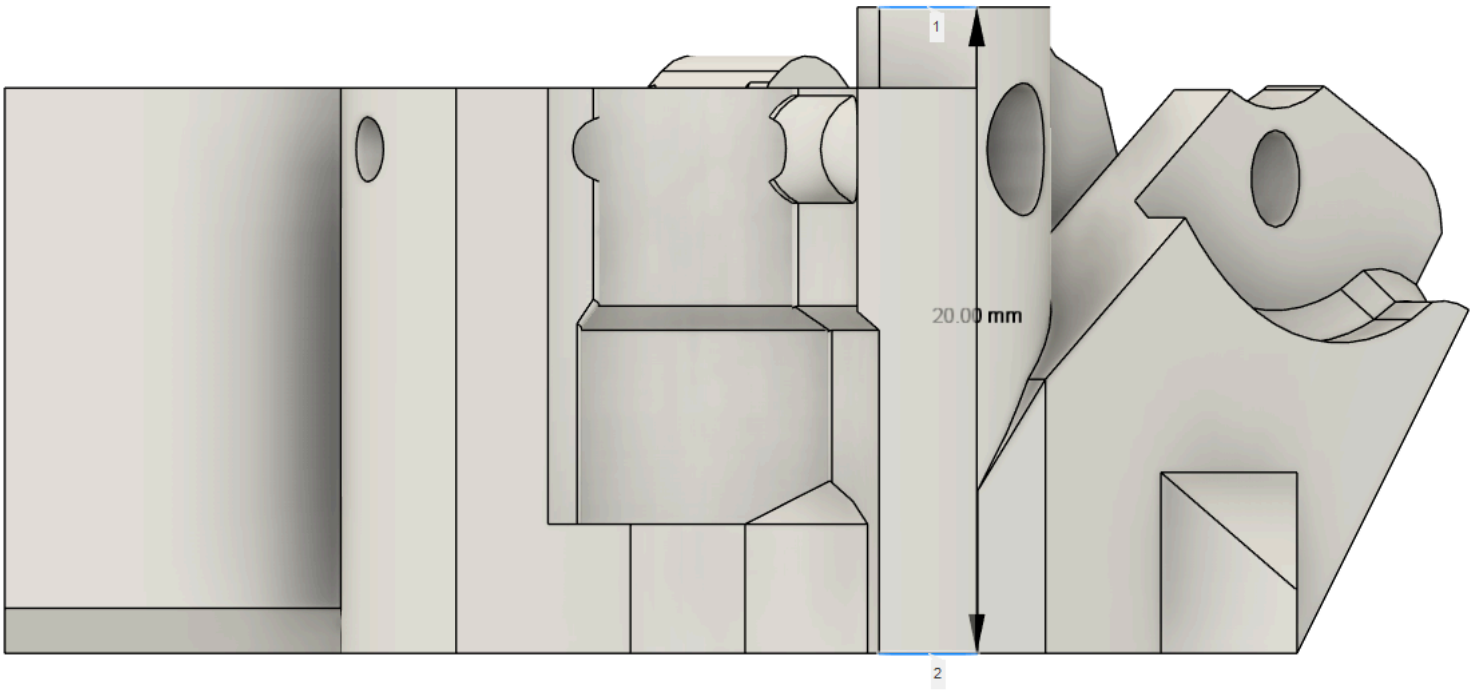
Test Point 1 – Model Z Height

Level of Importance: Critical

Measure at the shown location to confirm that your model height in the Z direction is identical to this value. Printed parts throughout Blackbox are designed to accept some level of deviation, but the printed parts for an FDM tool head will not align properly if not correct in the Z direction (as printed).

Pointers for a failed test print:

1. **Ensure you are slicing all Blackbox models at 0.2mm layer heights, INCLUDING the first layer.**
2. **Ensure your printers Z axis moves freely and is adjusted properly.**
3. **Ensure your first layer height shows minimal (if any) elephant's foot.**
4. **Ensure your extruders steps/mm are correct.**
5. **If all else is OK, you can consider adjusting your Z axis steps/mm.**



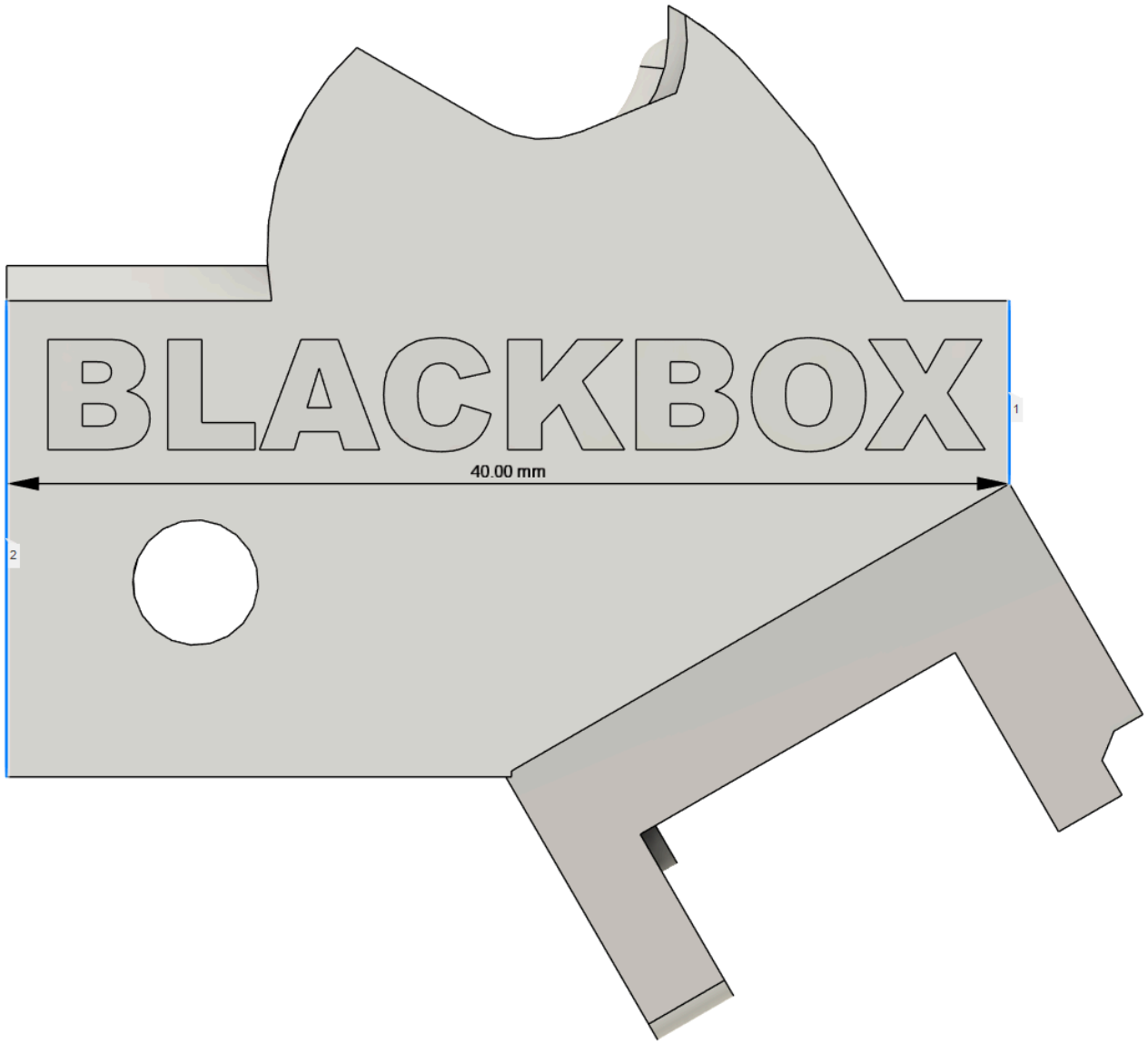
Test Point 2 – Model Y Width

Level of Importance: Moderate

Measure at the shown location to confirm that your model's width is equal to this dimension. Many factors go into tuning this dimension. Reference calibration cube guides for tips. NOTE: Avoid using horizontal expansion/XY size compensation to achieve this value.

Pointers for a failed test print:

1. **Ensure your extruders steps/mm are correct.**
2. **Ensure your slicer's flow rate is tuned.**
3. **Ensure you are not applying any horizontal expansion (AKA XY compensation) in your slicer.**
4. **If all else is OK, you can consider adjusting your Y axis steps/mm. NOTE: VERY rarely are Y steps/mm the proper way to adjust this value. See the teaching tech link above for help.**



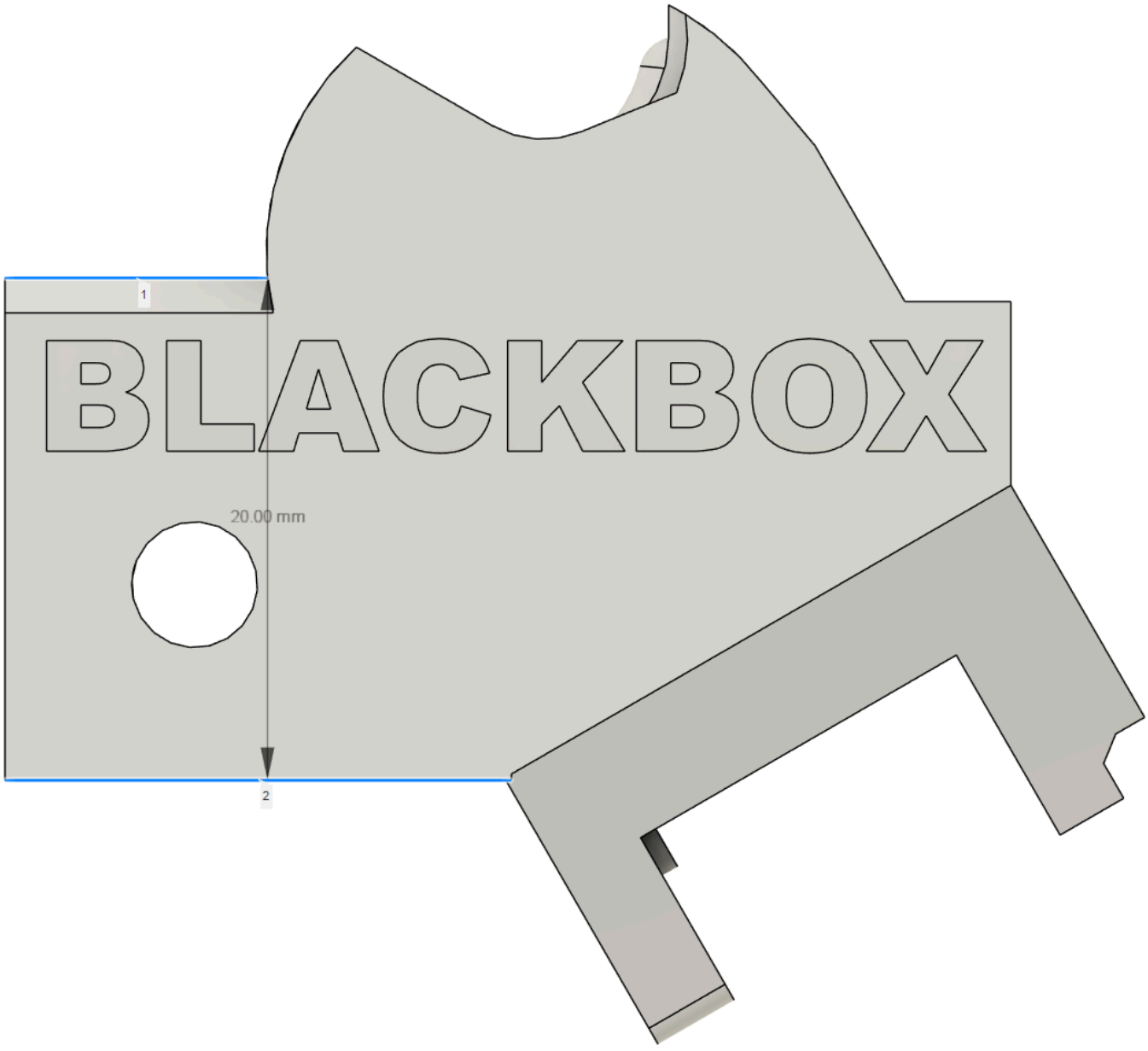
Test Point 3 – Model X Width

Level of Importance: Moderate

Measure at the shown location to confirm that your model's width is equal to this dimension. Many factors go into tuning this dimension. Reference calibration cube guides for tips. NOTE: Avoid using horizontal expansion/XY size compensation to achieve this value.

Pointers for a failed test print:

1. **Ensure your extruders steps/mm are correct.**
2. **Ensure your slicer's flow rate is tuned.**
3. **Ensure you are not applying any horizontal expansion (AKA XY compensation) in your slicer.**
4. **If all else is OK, you can consider adjusting your X axis steps/mm. NOTE: VERY rarely are X steps/mm the proper way to adjust this value. See the teaching tech link above for help.**

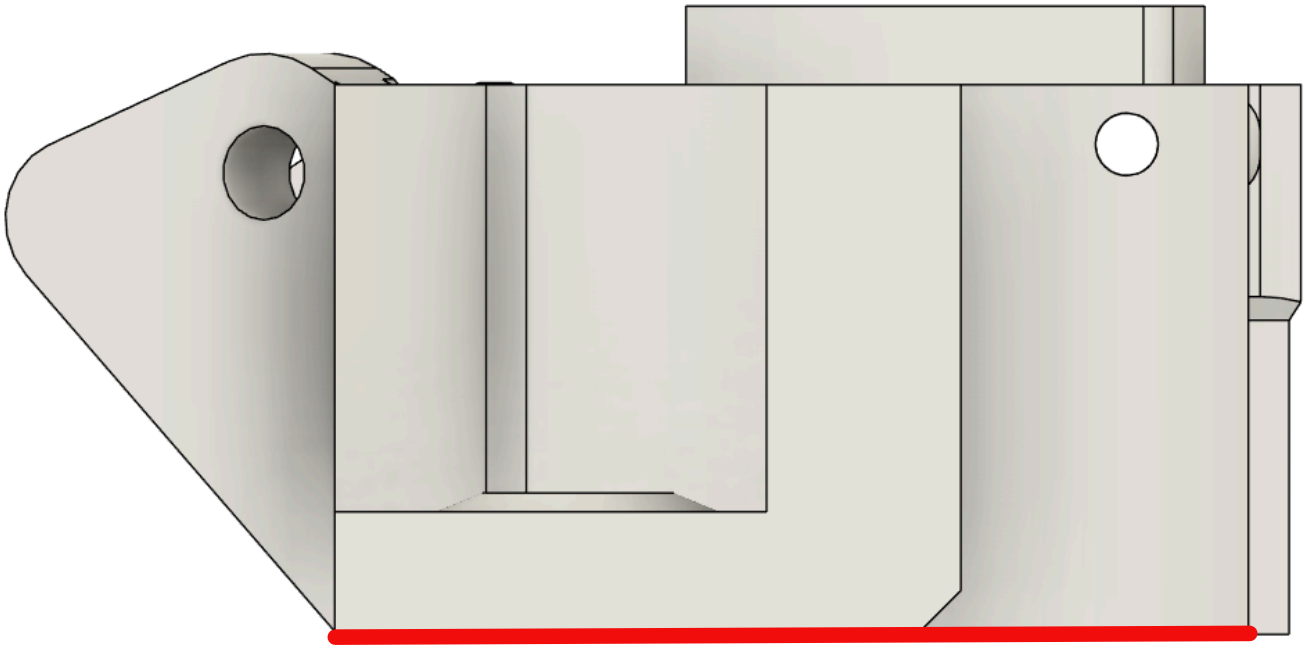


Test Point 4 – Elephants Foot

Level of Importance: Low

Elephant's foot refers to a bulge of material caused by the "squishing" of the first layer of extrusion. Some slicers have compensation settings for this. This can be removed manually with post-processing but will distort assemblies combining multiple printed parts if left untouched.

1. **95% of the time this issue is caused by a layer height that is too low. If you are struggling for adhesion, use some build plate adhesive.**
2. **The layers following a too-squished first layer will also tend to bulge, especially if they are solid layers.**
3. **Some slicers have a compensation tick box for this. It is always best to adjust your bed level and height to mitigate this issue instead.**



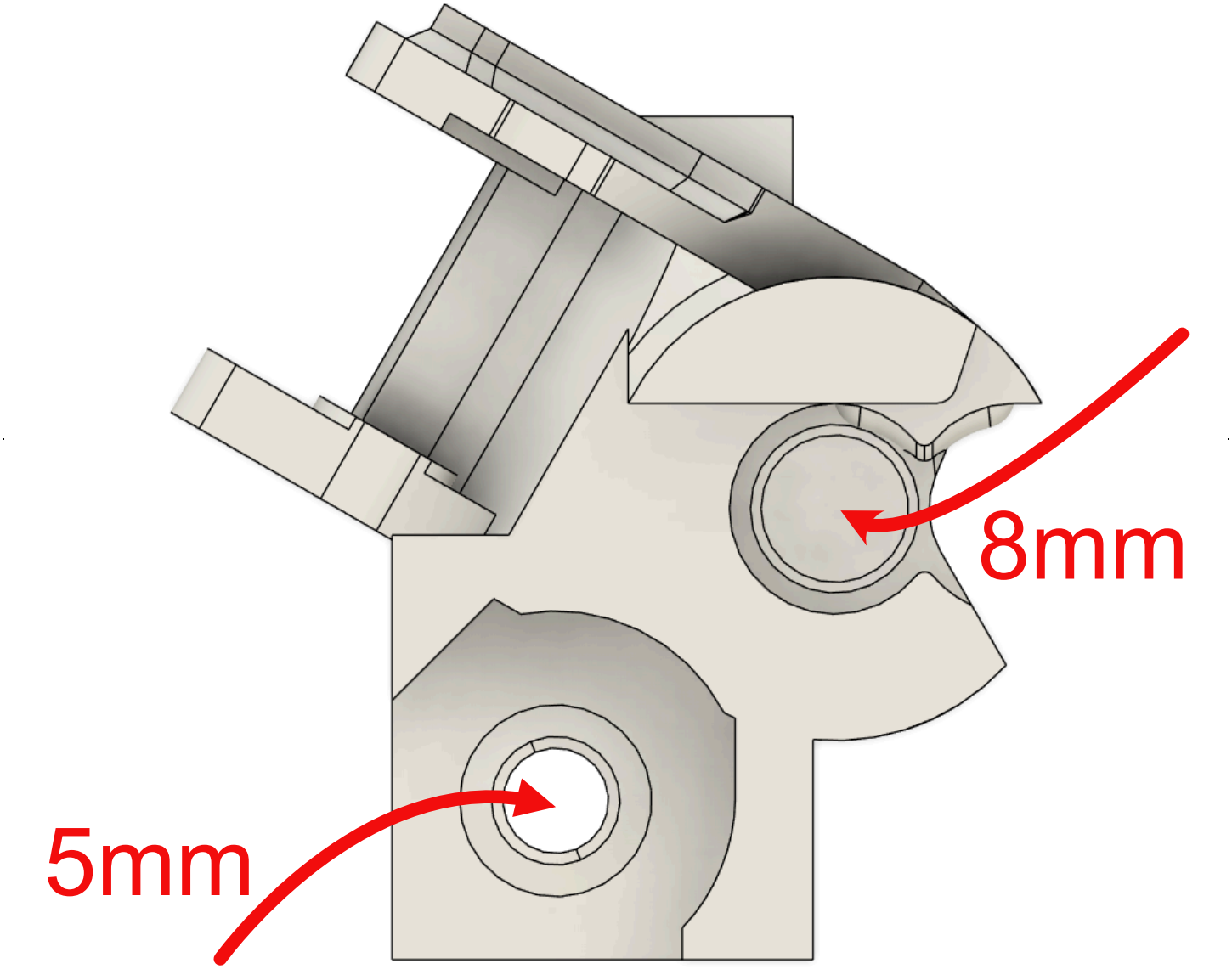
Test Point 6 – Horizontal Hole Diameter

Level of Importance: Moderate

Confirm the following dimensions of holes in the XY Plane. Typically, these holes will reflect the accuracy of the above X and Y dimensional tests. The 5mm hole shown should accept a 5mm dowel pin without excessive force. The 8mm bore shown should accept an 8mm OD Bondtech bearing without excessive force.

Pointers for a failed test print:

1. **Test points #2 and #3 above will directly affect these features.**
2. **Over or under extrusion will also affect bore diameters.**
3. **Always prioritize the results from test points #2 and #3, bore holes can be reamed if too small.**
4. **If bores are too large, stop and remedy this now.**



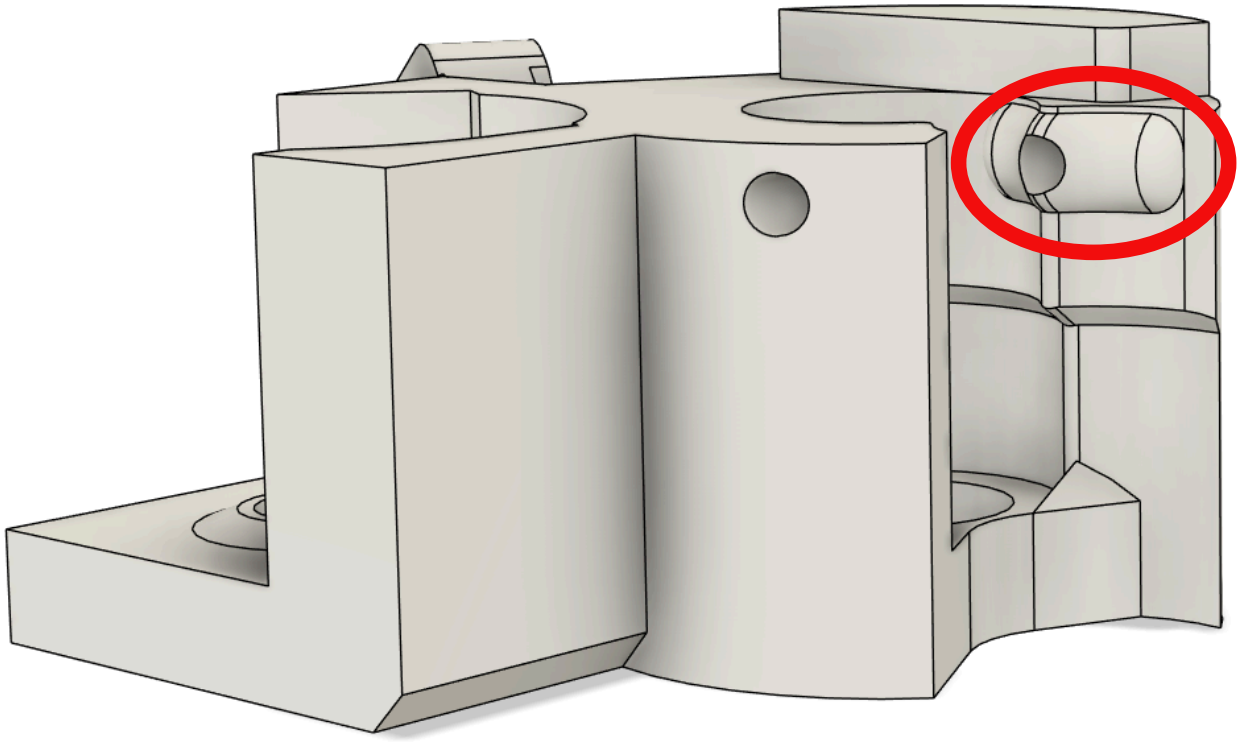
Test Point 7 – Detailed Overhangs

Level of Importance: Low - Moderate

Visual Inspection – This feature will serve as the filament constraint near the Bondtech drive gears. Poor reproduction of this feature will affect your extrusion rate ceiling when printing flexibles.

Pointers for a failed test print:

1. **Support roof interfaces help avoid sagging but can make support removal more difficult. This feature can print without support but will benefit from it.**
2. **Run your part cooling fan higher/faster when printing overhangs.**



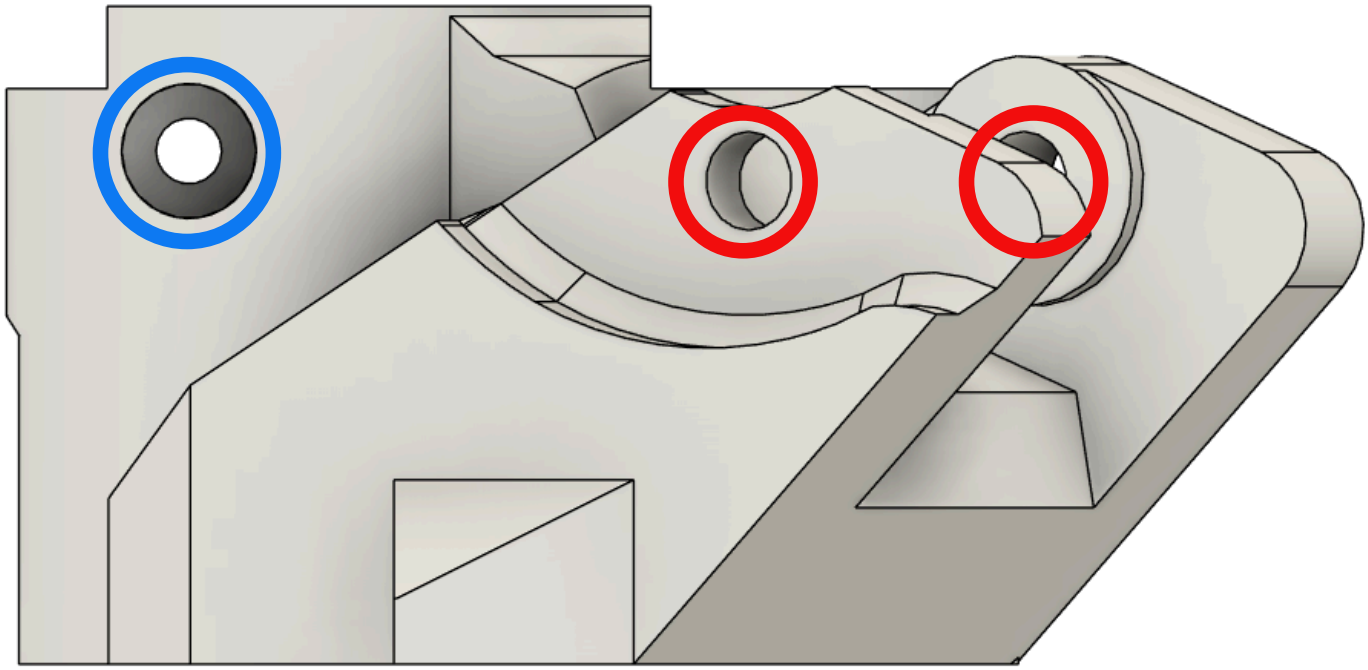
Test Point 8 – Vertical Holes

Level of Importance: Moderate

Vertical holes are always difficult to produce accurately in FDM printing. Blackbox printed parts are designed with this in mind. We highly recommend reaming all vertical holes to size with the appropriately sized reamer. The red (3mm) holes should accept a 3mm dowel. The blue (stepped 4mm, 2mm) bore should measure appropriately.

Pointers for a failed test print:

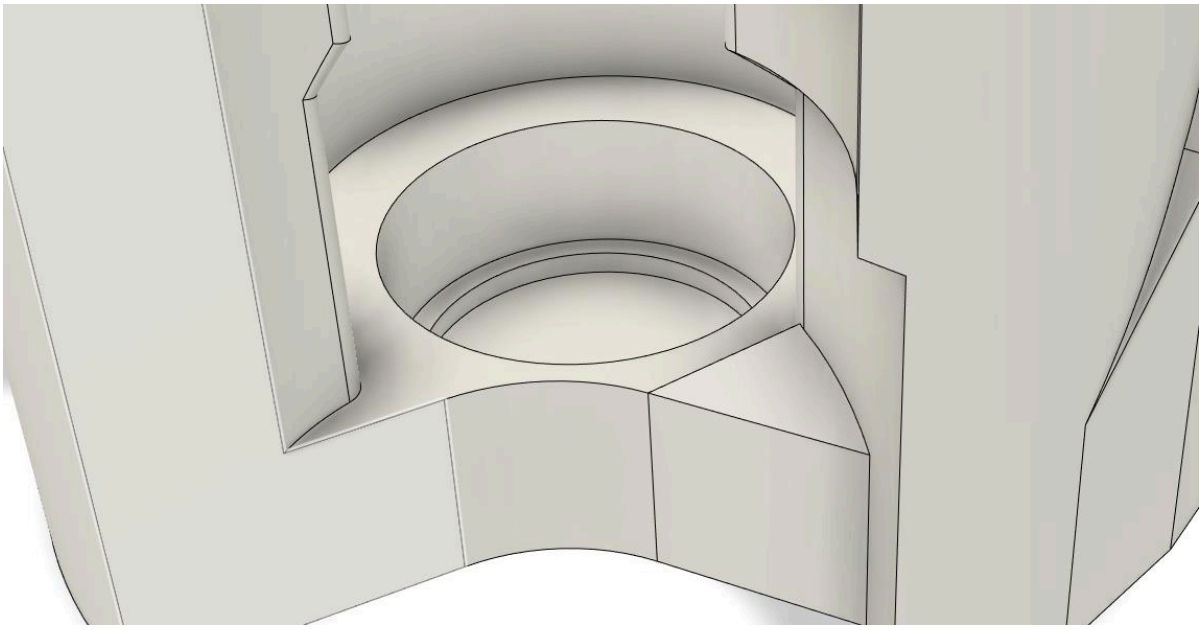
1. **Cura's tree supports excel with vertical hole features. Give them a try if you need to.**
2. **Run your part cooling fan higher/faster when printing overhangs.**
3. **Conventional supports can be used too, but you'll be aiming to find a balance between clean support and ease of removal.**



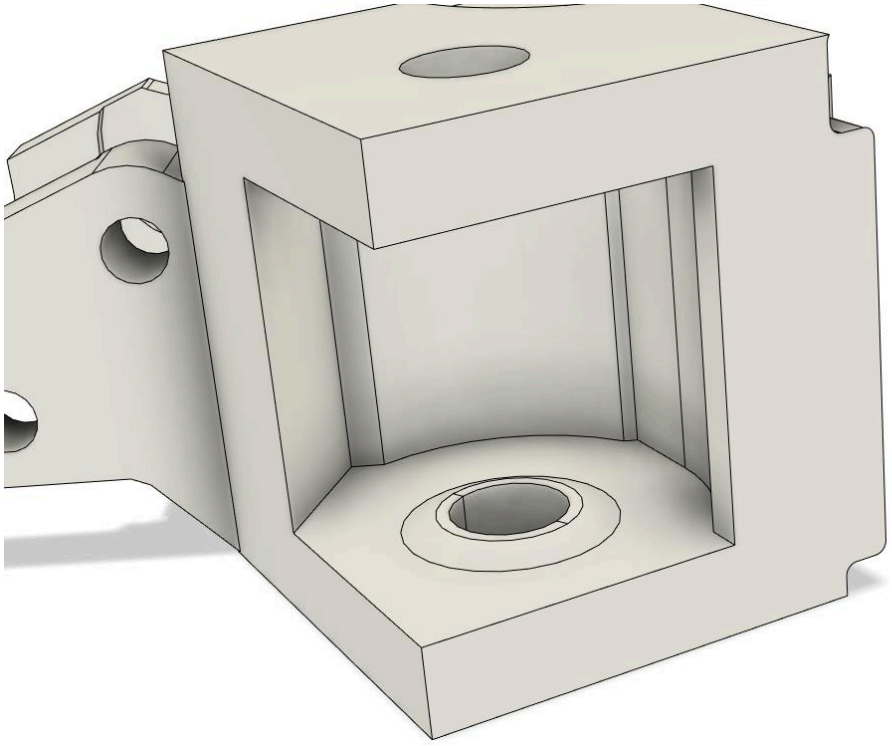
Features to Note:

While not necessarily needing to be measured, there are various other readiness indicators throughout this test print.

1. **Bearing Depth Setter – Ensure it prints nicely and is not damaged during support removal.**



2. **Sharp Corners – Ensure sharp corners do not exhibit signs of bulging. Linear advance can be tuned to resolve this.**



3. **Supported holes – These areas will normally require reaming but ensure that any generated support material is easily removable.**